SolidWorks Workshop
Institute of FIeld roBOtics

Shelling

Exercise 1: Clevis

Exercise 28: Clevis

Create this part using the dimensions provided. Use relations and equations where applicable to maintain the design intent.

This lab uses the following skills:

- Sketching.
- Shelling.

Or-

- Thin Features.

Units: **millimeters**

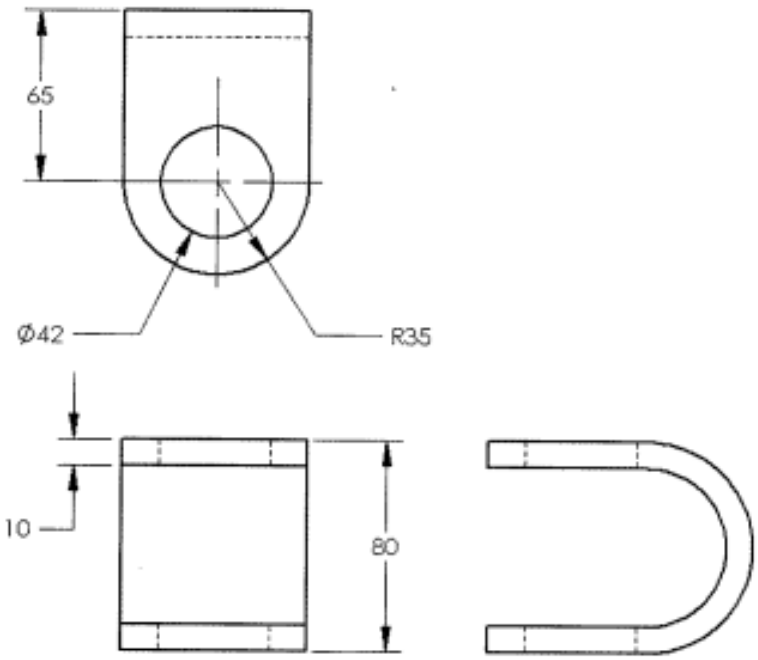
Design Intent

The design intent for this part is as follows:

1. Material thickness is constant.
2. Part is symmetrical.
3. Round holes are equal diameter and placement.
4. All fillets and rounds **2mm**.

Dimensioned Views

Use the following graphics with the design intent to create the part.

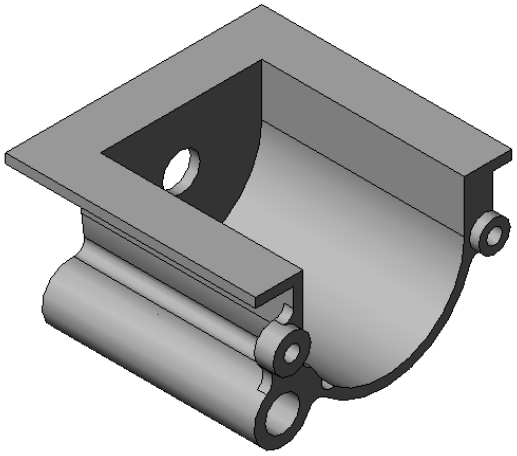


Exercise 2: Motor Shield (Shelling, Offsetting, Converting Entities)

Create this part using the step by step instructions provided. Use relations or link values where applicable to maintain the design intent.

This lab uses the following skills:

- Sketching.
- Shelling.
- Extrusions.



Design Intent

The design intent for this part is as follows:

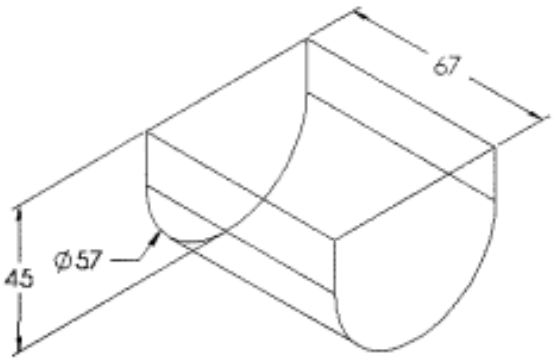
- Thickness is the same for `Wall Thick`, `Face` and `Top plate` features.

Procedure

Open a new part using the `Part_MM` template.

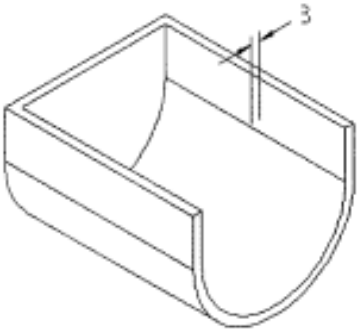
1 `Center body` feature.

Sketch the profile on the `Right` plane and extrude it.

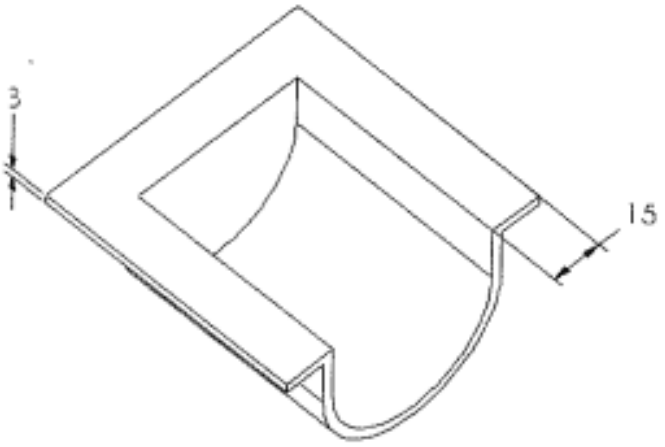


2 `Wall Thick` feature.

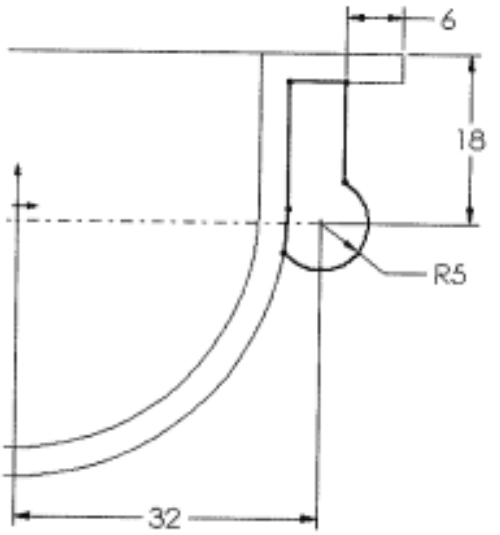
Shell the solid, removing two faces and shelling to the *inside*.



3 **Top plate feature.**
Sketch using converted and offset model edges. Extrude *into* the model.

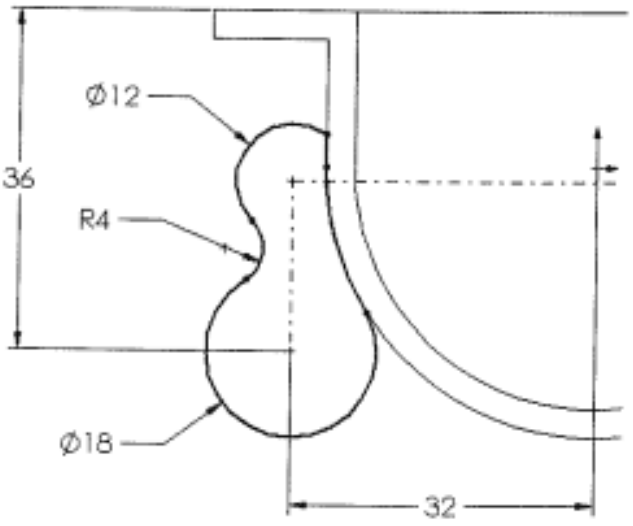


4 **Face sketch — rightmost portion.**
Using converted edges and sketch geometry, sketch the profile. Extrude the sketch *into* the solid **3mm**.



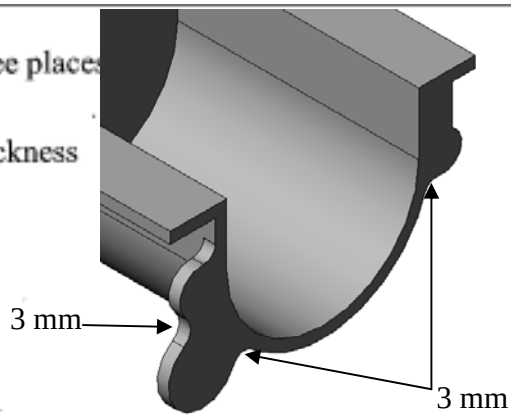
5 **Face sketch — leftmost portion.**
Using converted edges and sketch geometry, sketch the profile. Extrude the sketch into the solid the same depth as the previous feature.

The center of the **12mm** diameter arc is aligned horizontally with the center of the **5mm** radius arc shown in step 4.

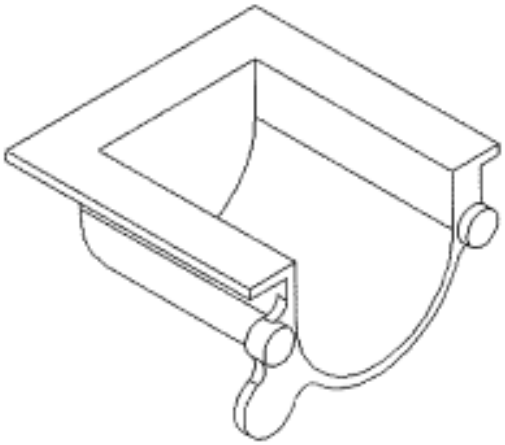


6 **Fillets.**
Add fillets of radius **3mm** in three places

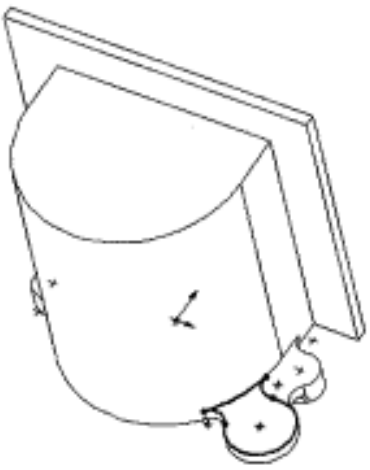
7 **Link Values.**
Use **Link Values** to keep the thickness dimensions (**3mm**) equal.



8 **`Circular` boss feature.**
Using existing edges of the model, create two circular bosses **Coradial**. Extrude the sketch away from the solid **3mm**.

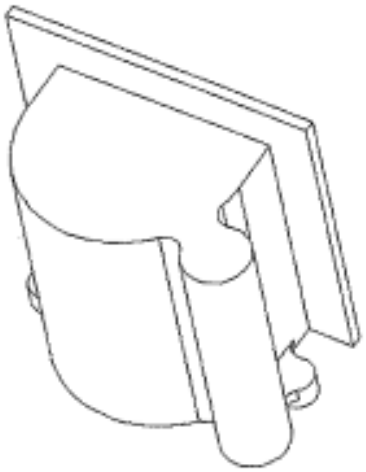


9 **`Extension` sketch.**
Copy edges of the model and add sketch geometry to create the sketch. Both radius values are equal.



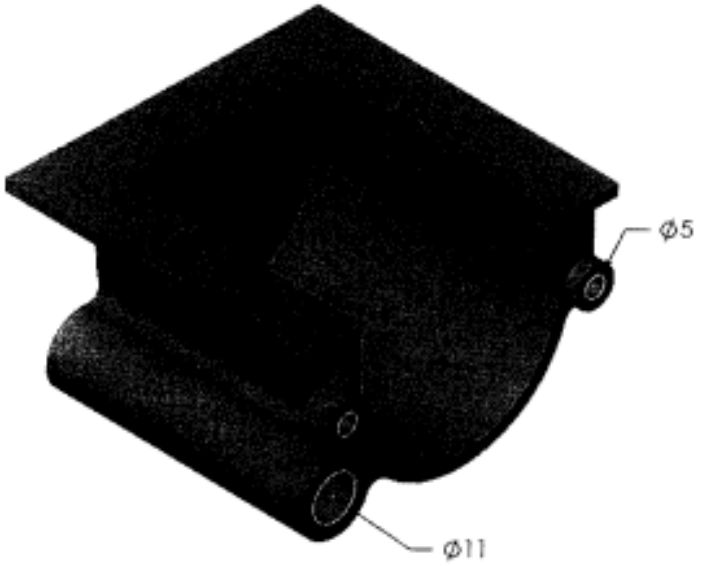
10 `Extension` feature.

Use the **Up To Surface** end condition for this extrusion up to the rear face of the solid.



11 `Thru hole` feature.

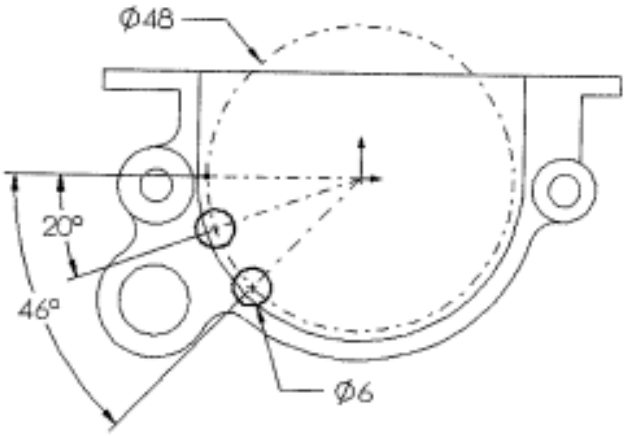
Create the `Thru hole` feature as one or more features. Both smaller holes have the same diameter.



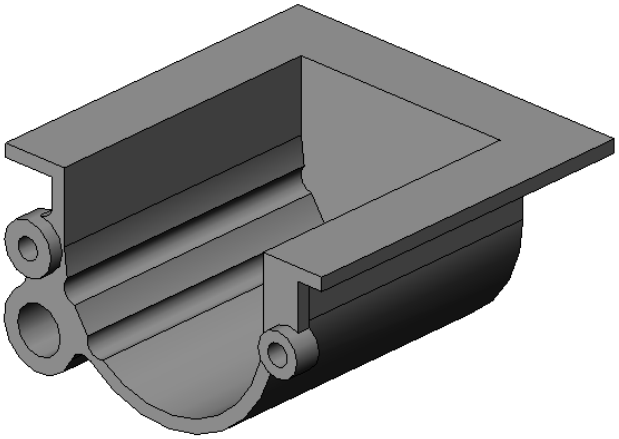
12 `Arc cut` sketch.

Construct the geometry for the `Arc cut` sketch using construction geometry, converted edges and sketch geometry.

Circular construction geometry is created normally and changed using **Properties**.



13 `Arc cut` feature.
Create features that cut into the model along the inside faces.



14 `Center hole` feature.
Create the `Center hole` as a **Through All** cut.

15 Save and close the part.